

Comparison of Riveted Panel and Integral Panel for Damage Tolerance by Using FEA

Dr. Shantharaja M.¹, Srinivas K. R.²

¹Asst. Professor, Dept of Mechanical Engineering, UVCE, K.R. Circle, Bangalore 560001

²4th sem, M.E Student, UVCE, K.R. Circle, Bangalore 560001

Abstract: Damage tolerance capability refers to the load carrying capacity of a damaged structure. In an aircraft metallic structure damage is the fatigue crack. The presence of a fatigue crack reduce the load carrying capacity of a structure. Damage tolerance design ensures that a structural component, in the presence of a fatigue crack, continue to possess the required load carrying capacity. Here the role of the type of the structural joints (riveted and integral) on damage tolerance quality of a structural component will be studied.

Keywords: Riveted panel, Integral panel, Stress concentration factor and Stress intensity factor

1. Introduction

Aircraft members and transverse frames to enable it to resist bending, compressive and vehicles which are able to fly by being supported by the air, or in general, the atmosphere of a planet. The main body structure is the fuselage to which all other components are attached and it is to be pressurized with increase in altitude. Due to pressurization of the fuselage, the stresses will be induced in the structure. These is studied by introducing the crack of different length by using FEA.

2. Finite element analysis

The stress analysis of Fuselage of the Transport aircraft has been carried out using

- MSC NASTRAN
- MSC PATRAN

2.1 MSC NASTRAN

MSC NASTRAN is one of the most popular general purpose finite element packages available for structural analysis. The package can support linear static/dynamic/buckling analysis of general composite structures under thermo-mechanical loads. The element library of the package is quite extensive including mainly linear elements (e.g. BAR, QUAD4, HEXA8 etc.) A flowchart summarizing the basic steps in MSC NASTRAN linear static structural analysis is shown in Figure 1.

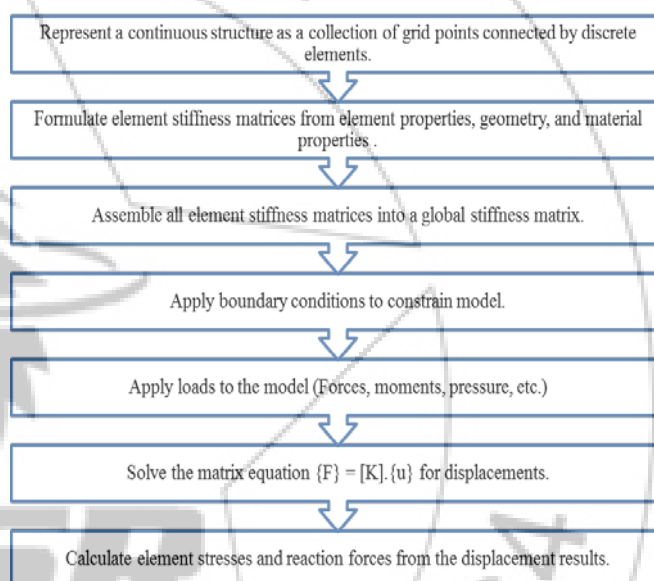


Figure 1: NASTRAN Solution Flow Chart

2.2 MSC PATRAN

Meshing is done in MSC Patran by using various meshing options. Good quality of the mesh can be achieved from making use of options like element density, type, biasing, smoothing, shaping, sizing, etc. Once elements are created, care should be taken to organize them to respective collectors, as the elements behave according to the property given to the collector.

2.3 Material Properties

Table 1: Material Properties

Material	Properties				
	E MPa	γ	σ_t MPa	σ_y MPa	ρ g/cm ³
Aluminum 2024-T3	68670	0.3	441.2	338	2.78

3. Stress Analysis of Riveted Stiffened Panel

3.1 Finite element model of the riveted stiffened panel

Finite element meshing is carried out for all the components of the stiffened panel. Rivets are simulated using beam elements. Fine meshing is done at the critical sections where stresses are expected to be more. The following figures show the details about the finite element mesh generated on each part of the structure using MSC PATRAN.

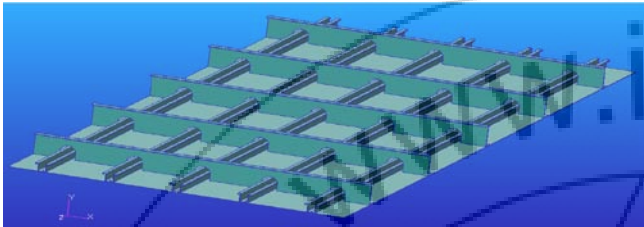


Figure 2: Finite element model of the riveted stiffened panel

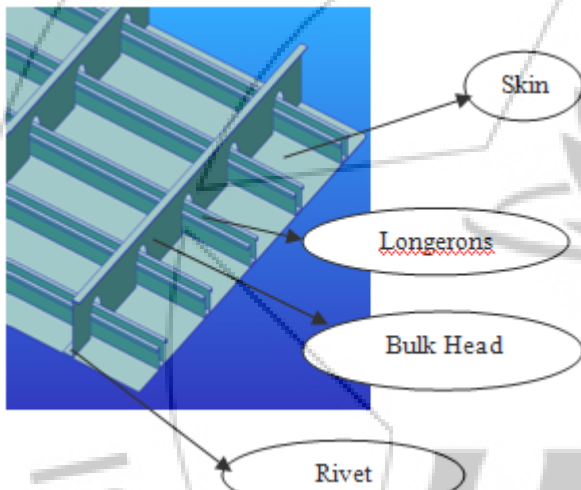


Figure 3: Closed view of riveted stiffened panel

3.2 Loads in the stiffened panel

A differential pressure of 0.0413688MPa is considered for the current case. Due to this internal pressurization of fuselage (passenger cabin) the hoop stress will be developed in the fuselage structure. The tensile loads at the edge of the panel corresponding to pressurization will be considered for the linear static analysis of the panel. Hoop stress is given by

$$\sigma_{\text{hoop}} = \frac{Pxr}{t}$$

Where

Cabin pressure (P) = 0.0413688 MPa

Radius of curvature of fuselage (r) = 750 mm

Thickness of skin (t) = 2 mm

$$\sigma_{\text{hoop}} = 15.5 \text{ MPa}$$

We know that,

$$\sigma_{\text{hoop}} = \sigma_{\text{tension in skin}}$$

$$\sigma_{\text{tension in skin}} = \frac{P}{A}$$

$$\left(\frac{Pxr}{t}\right)_{\text{hoop stress}} = \left(\frac{P}{A}\right)_{\text{tension in skin}}$$

3.2.1 Uniformly distributed tensile load is applied on the stiffened panel in X axial direction

Load on the skin

Here

$$P_s = \text{Load on skin N}$$

$$\sigma_{\text{hoop}} = 15.5 \text{ MPa}$$

$$A = \text{Cross sectional area of skin in mm}^2$$

$$\text{i.e. Width X Thickness (2250 X 2) = 4500 mm}^2$$

Substituting these values, we get

$$P_s = 69750 \text{ N}$$

Uniformly distributed load on skin will be

$$P_s = 69750 / 2250 = 31 \text{ N/mm}$$

Load on Bulkhead

Here

$$P_b = \text{load on Bulkhead in N}$$

$$\sigma_{\text{hoop}} = 15.5 \text{ MPa}$$

$$A = \text{Cross sectional area of each Bulkhead in mm}^2$$

$$\text{i.e. Width X Thickness, (H1+H2+H3) X t_b}$$

$$\text{i.e. (20 + 78.2 + 28.2) X 1.8 = 227.52 mm}^2$$

$$\left(\frac{Pr}{t}\right)_{\text{hoop stress}} = \left(\frac{P}{A}\right)_{\text{tension in bulkhead}}$$

$$15.5 = \left(\frac{P}{227.52}\right)$$

$$P_b = 3526.56 \text{ N}$$

Uniformly distributed load on Bulkhead will be

$$P_b = \frac{3526.56}{126.4} = 27.9 \text{ N/mm}$$

3.3 Loads and boundary conditions of the riveted stiffened panel

All the edge nodes of stiffened panel are constrained in all five degree of freedom (i.e. 23456) except loading direction which is X direction (i.e. 1). All the elements along the thickness direction are constrained to avoid the eccentricity due to stiffening members.

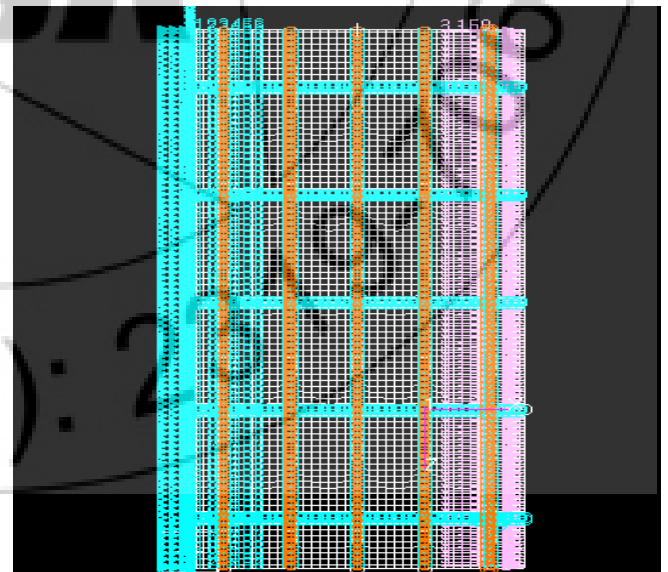


Figure 4: Loads and boundary conditions of the riveted stiffened panel

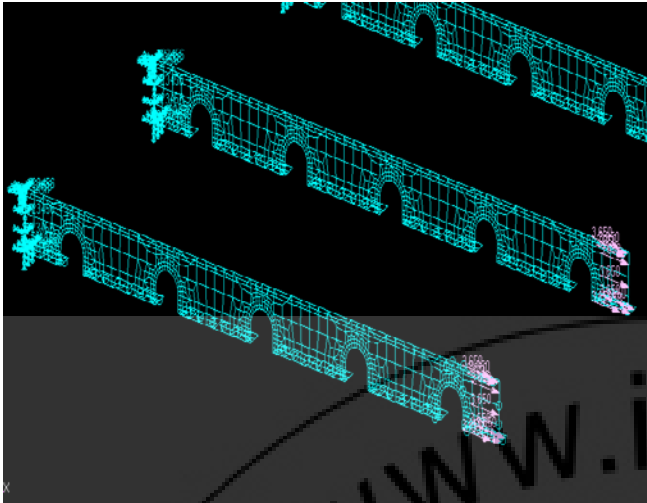


Figure 5: Loads and boundary conditions on Bulkhead

3.4 Results obtained from the finite element analysis of the riveted stiffened panel.

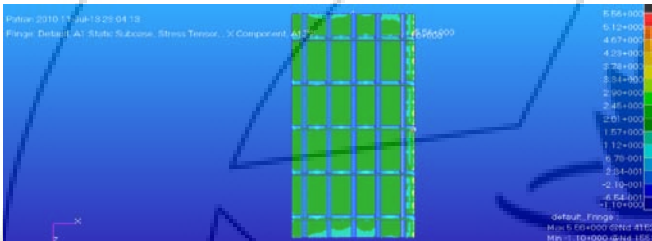


Figure 6: Stress counter of the riveted stiffened panel

Fig 6 shows the stress contour on the stiffened panel from global analysis results. It is clear that the maximum tensile stress on stiffened panel is at stringer cut-out (mouse cut-out) and this maximum tensile stress is uniform in all the stringer cut-outs.

4. Stress Analysis of Integral Stiffened Panel

Finite element meshing is carried out for all the components of the stiffened panel. Fine meshing is done at the critical sections where stresses are expected to be more. The following figures show the details about the finite element mesh generated on each part of the structure using MSC PATRAN.

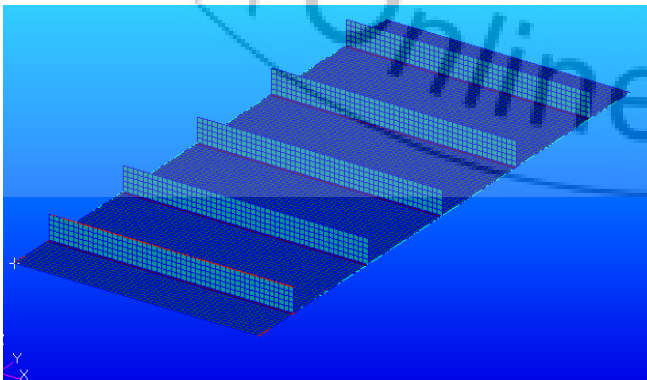


Figure 7: Finite element model of the integral stiffened panel

4.1 Loads in the stiffened panel

A differential pressure of 0.0413688MPa is considered for the current case. Due to this internal pressurization of fuselage (passenger cabin) the hoop stress will be developed in the fuselage structure. The tensile loads at the edge of the panel corresponding to pressurization will be considered for the linear static analysis of the panel.

Hoop stress is given by

$$\sigma_{\text{hoop}} = (0.0413688 \times 750) / (2) = 15.5 \text{ MPa}$$

1) Uniformly distributed tensile load is applied on the stiffened panel in X axial direction.

Load on the skin

Here, P_s = Load on skin

$$\sigma_{\text{hoop}} = 15.5 \text{ MPa}$$

A = Cross sectional area of skin in mm^2

i.e. Width X Thickness $(2250 \times 2) = 4500 \text{ mm}^2$

Substituting these values in the Eq-5.2 we get

$$P_s = 69750 \text{ N}$$

Uniformly distributed load on skin will be

$$P_s = 69750 / 2250 = 31 \text{ N/mm}$$

Load on Bulkhead

Here, P_b = load on Bulkhead in Kg

$$\sigma_{\text{hoop}} = 15.5 \text{ MPa}$$

A = Cross sectional area of each Bulkhead in mm^2

i.e. Width X Thickness, $(H1 + H2) \times t_b$

i.e. $(82 + 32) \times 2 = 228 \text{ mm}^2$

Substituting these values, we get

$$15.5 = \left(\frac{P}{228} \right)$$

$$P_b = 3534 \text{ N}$$

Uniformly distributed load on Bulkhead will be

$$P_b = \frac{3534}{114} = 31 \text{ N/mm}$$

The load on the flange bulkhead for integral panel

= load acting on skin + load acting on bulkhead

$$= 31 + 31$$

Total load acting on the flange bulkhead = 62 N/mm

4.2 Loads and boundary conditions of the integral stiffened panel

All the edge nodes of stiffened panel are constrained in all five degree of freedom (i.e. 23456) except loading direction which is X direction (i.e. 1). All the elements along the thickness direction are constrained to avoid the eccentricity due to stiffening members.

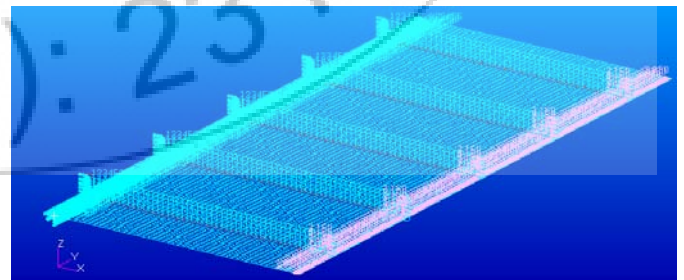


Figure 8: Loads and boundary conditions of the integral stiffened panel

4.3 Results obtained from the finite element analysis of the integral stiffened panel

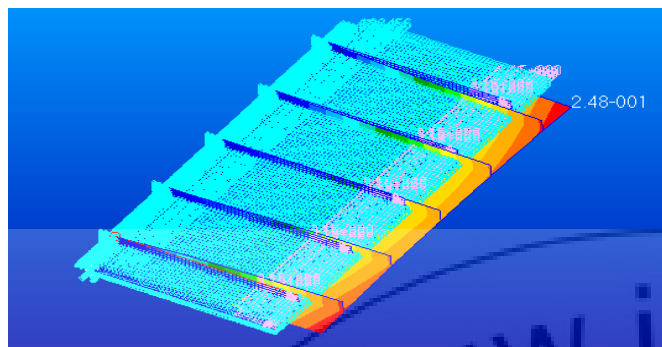


Figure 9: Displacement contour of the integral stiffened panel

Fig 9 shows the stress contour on the skin from the integral stiffened panel analysis results. It is clear that the maximum stress on skin is at the rivet location where the rivets are used to fasten the bulkheads and skin. The maximum stress locations are the probable locations for crack initiation. Invariably these locations will be at rivet locations in the skin.

5. Damage Tolerance Evaluation of Riveted Stiffened Panel

5.1 MVCCI method for calculation of SIF

The MVCCI is based on the energy balance proposed by Irwin. In this technique, SIF is obtained for first fracture mode from the equation.

$$G_i = \frac{K_i^2}{E} \beta \quad (i=1,2,3)$$

Where G_i is the energy release rate for mode i , K_i the stress intensity factor for mode i , E the elastic modulus, ν the Poisson ratio, $\beta=1$ for plane stress, and $\beta=1-\nu^2$ for plane strain. Calculation of the energy release rate is based on Irwin assumption that the energy released in the process of crack expansion is equal to work required to close the crack to its original state as the crack extends by a small amount Δa . Irwin computed this work as

$$W = \frac{1}{2} \int_0^{\Delta a} u(r) \sigma(r - \Delta a) dr,$$

Where u is the relative displacement, σ the stress, r the distance from the crack tip, and Δa the change in virtual crack length. After simplification, Strain energy release rate.

$$G = \frac{1}{2\Delta a} \times \frac{F}{t} \times \Delta V$$

Where,

Δa = Element length near the crack tip

ΔV = Displacement of nodes near the crack tip

F = Force at the crack tip

t = Thickness of the skin

5.2 Analysis of riveted panel

Analysis of riveted panel can be done by the considering the rectangular plate (2250*1000mm) with center crack. By

varying the crack lengths keeping the plate dimensions and load constant, SIF for different crack length are obtained.

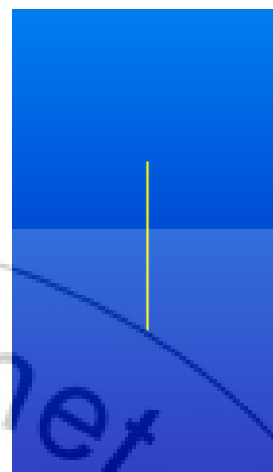


Figure 10: Plate with center crack

5.3 Loads and boundary conditions

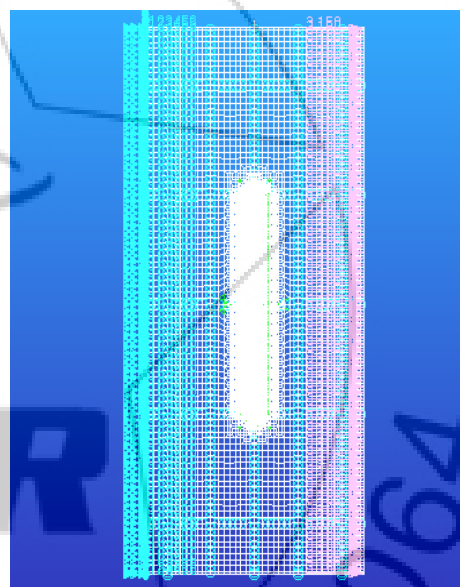


Figure 11: Load and boundary condition in unstiffened panel

All the edge nodes of plate are constrained in all five degree of freedom (i.e.23456).except loading direction which is X direction (i.e. 1). At the loading direction(X direction) UDL (uniformly distributed load) is applied. All the elements along the thickness direction are constrained to avoid the eccentricity due to stiffening members.

5.4 Results obtained from the finite element analysis of the riveted plate with center crack

5.4.1 SIF Calculation

SIF for LEFM methods

$$SIF = K_I = \sigma_r \times \sqrt{(\pi a)} \quad \text{MPa}\sqrt{\text{m}}$$

$$\sigma_r = \frac{P}{A} = \left(\frac{69750}{2250 \times 2} \right) = 15.5 \text{ MPa}$$

$$SIF = K_I = 15.5 \times \sqrt{(\pi 200 \times (10)^{-3})} = 12.28 \text{ MPa}\sqrt{\text{m}}$$

For Crack Length in mm ($2a=400$, $a=200$)

SIF for MVCCI methods

$$K_I = \sqrt{(GE)} \dots \text{MPa} \sqrt{\text{m}}$$

$$G = \frac{1}{2\Delta a} \times \frac{F}{t} \times \Delta V$$

Substituting the values we get

$$G = 2.878 \text{ N/mm}$$

$$E = 68670 \text{ N/mm}^2$$

$$K_I = \sqrt{(0.291406 \times 68670)} = 14.04 \text{ MPa} \sqrt{\text{m}}$$

From the analysis of riveted panel for different crack length, we got node displacement and force acting on the respective node and is used for calculating SIF and energy release rate for different crack length varying from 100mm to 900mm. The variation of SIF along with crack length is plotted below.

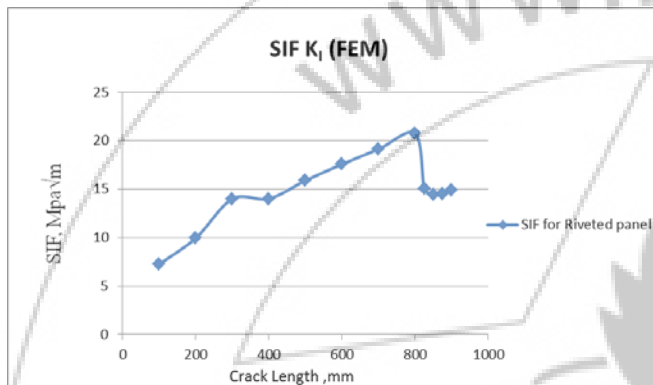


Figure 12: SIF v/s crack length

In this graph SIF for different crack length is plotted. SIF increases gradually with increase in the crack length. As the crack reaches nearer to the stiffening member (Riveted bulkhead) there is noticeable decrease in SIF, this may be due to transfer of load from the skin to bulkhead. This interprets the crack arrests capability of the stiffening member.

6. Damage Tolerance Evaluation of Integral Stiffened Panel

The Modified Virtual Crack Closure Integral method (MVCCI) is used to find the value of stress intensity factor and strain energy release rate for integral panel.

6.1 Loads and boundary conditions

All the edge nodes of plate are constrained in all five degree of freedom (i.e. 23456) shown in fig 13. except loading direction which is X direction (i.e. 1). At the loading direction (X direction) UDL (uniformly distributed load) is applied shown in fig.14. All the elements along the thickness direction are constrained to avoid the eccentricity due to stiffening members.

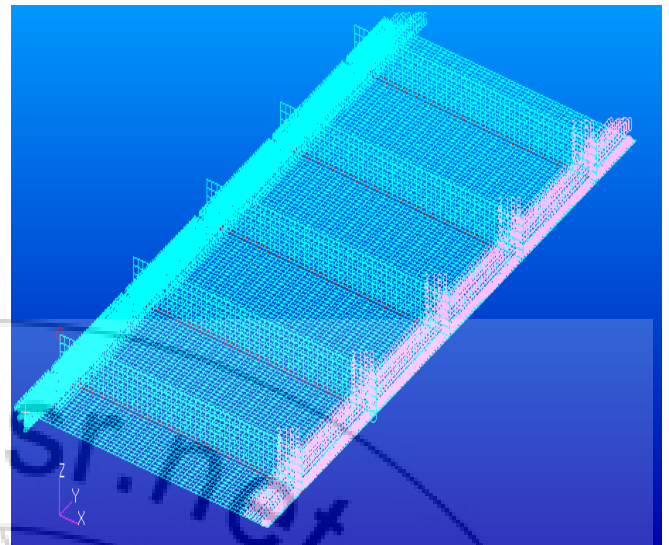


Figure 13: Load and boundary condition in skin and bulkhead

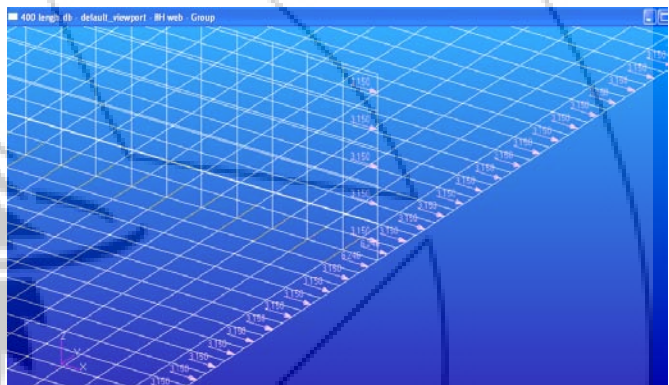


Figure 14: Close view of UDL distribution

6.2 Results obtained from the finite element analysis of the integral plate with center crack

6.2.1 SIF Calculation

SIF for LEFM methods

$$SIF = K_I = \sigma_r \times \sqrt{(\pi a)}$$

$$\sigma_r = \frac{P}{A} = \left(\frac{69750}{2250 \times 2} \right) = 15.5 \text{ MPa}$$

$$SIF = K_I = 15.5 \times \sqrt{(\pi 200 \times (10)^{-3})} = 12.28 \text{ MPa} \sqrt{\text{m}}$$

FOR CRACK LENGTH OF MM (2a=400, a=200)

SIF for MVCCI methods

$$K_I = \sqrt{(GE)} \dots \text{MPa} \sqrt{\text{m}}$$

Strain energy release rate is given by

$$G = \frac{1}{2\Delta a} \times \frac{F}{t} \times \Delta V$$

Substituting the values we get

$$G = 3.307 \text{ N/mm}$$

$$E = 68690 \text{ N/mm}^2$$

$$K_I = \sqrt{(0.33436 \times 68690)} = 15.05 \text{ MPa} \sqrt{\text{m}}$$

From the analysis of integral panel for different crack length, we got node displacement and force acting on the respective node and is used for calculating SIF and energy release rate for different crack length varying from 100mm to 900mm. The variation of SIF v/s crack length is plotted below.

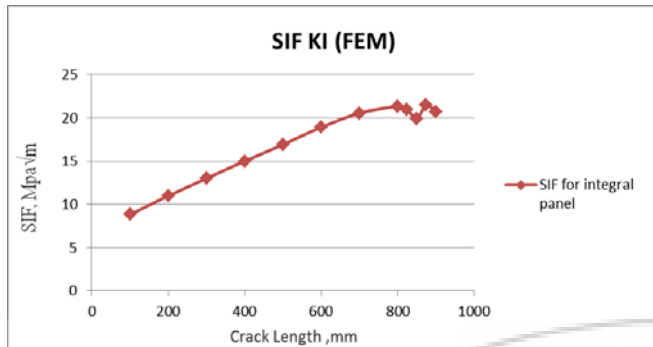


Figure 15: SIF v/s crack length

In this graph SIF for different crack length is plotted. SIF increases gradually with increase in the crack length. As the crack reaches nearer to the stiffening member (Integral bulkhead) there is slightly decrease in SIF.

6.3 Comparison of SIF for riveted panel v/s integral panel

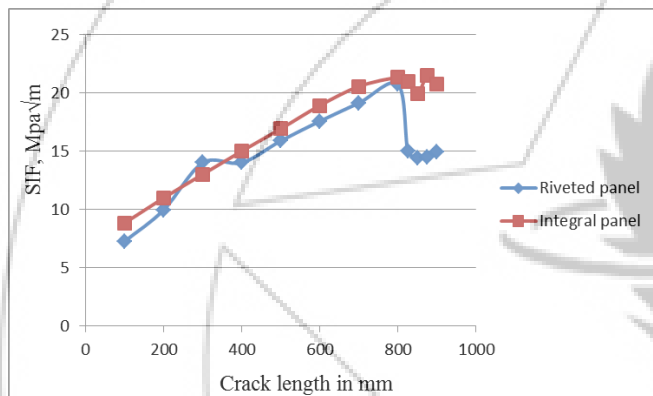


Figure 16: SIF for riveted panel v/s integral panel

In this graph SIF compare both riveted panel and integral panel, SIF in integral panel is more than the riveted panel.

7. Conclusion

SIF increases gradually with increase in the crack length. As the crack reaches nearer to the stiffening member (bulkhead) there is noticeable decrease in SIF of riveted panel compared to integral panel, this is due to transfer of load from skin to the bulkhead in riveted panel. The SIF of riveted panel is less than the integral panel as shown in Fig.16.

References

- [1] Venkatesh B.K., Suresh B.S. Girish K.E., "Analytical Evaluation of Fatigue Crack Arrest Capability in Fuselage of Large Transport Aircraft" ISSN (Print): 2319-3182, Volume-1, Issue-1, 2012.
- [2] Ankur Joshi, Lakshmi Prasad and Girish K.E "Damage tolerance evaluation of a stiffened panel with a passenger window cutout of a transport airframe fuselage structure" 2014 IJEDR | Volume 2, Issue 2 | ISSN: 2321-9939.
- [3] H. Vlieger, "The Residual Strength Characteristics Of Stiffened Panels Containing Fatigue Cracks" National Aerospace Laboratory NLR. Amsterdam. The Netherlands, 1973. Vol. 5. pp. 447-477.

- [4] J Schijve, "Fatigue Damage In Aircraft Structures, Not Wanted, But Tolerated" Delft University of Technology, Faculty of Aerospace Engineering Kluyverweg , 2629 HS, The Netherlands.
- [5] Dr. M. M. Nadakatti, Vinayakumar. B. Melmari, "A Review on Stress Analysis of the Fuselage Structure and Study of the Effect of Overload on Fatigue Crack Growth", International Journal of Engineering and Innovative Technology (IJEIT) Volume 1, Issue 4, April 2012.
- [6] S.M.O. Tavares1 , P.M.S.T. de Castro, "Stress Intensity Factor Calibration For A Longitudinal Crack In A Fuselage Barrel" Faculdade de Engenharia da Universidade do Porto and IDMEC-Porto Rua Dr. Roberto Frias, 4200-465 Porto, Portugal.
- [7] Shamsuzuha Habib, K.S. Raju, "Crack Arrest Capabilities in Adhesively Bonded Skin and Stiffener" Department of Aerospace Engineering, College of Engineering, GRASP Symposium, Wichita State University, 2009.
- [8] X Zhang, M Boscolo, D Figueroa-Gordon, G Allegri, PE Irving, "Fail-Safe Design of Integral Metallic Aircraft Structures Reinforced by Bonded Crack Retarders" Departments of Aerospace Engineering and Materials, Cranfield University Bedfordshire, MK43 0AL, U.K. *Engineering Fracture Mechanics* 76 (2009): 114-133. doi:10.1016/j.engfractmech.2008.02.003.